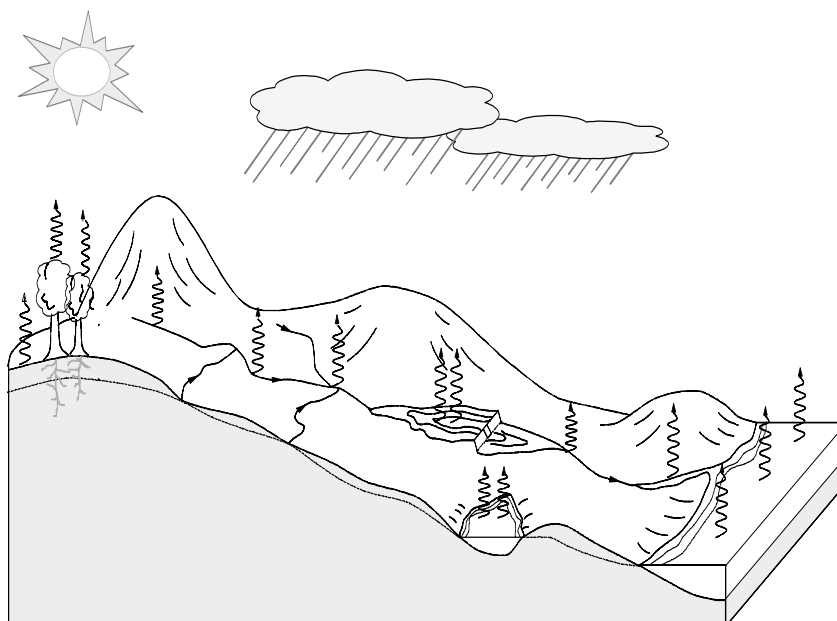**US Army Corps of Engineers** ®
Hydrologic Engineering Center

Hydrologic Modeling System HEC-HMS

Quick Start Guide DRAFT

Version 3.0
June 2005

CPD-74D

REPORT DOCUMENTATION		Form Approved OMB No. 0704-0188

Public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden, to Washington Headquarters Services, Directorate for Information Operations and Reports, 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302, and to the Office of Management and Budget, Paperwork Reduction Project (0704-0188), Washington DC 20503.

1. AGENCY USE ONLY *(Leave Blank)*	2. REPORT DATE June 2005	3. REPORT TYPE AND DATES COVERED Computer Software User's Manual

4. TITLE AND SUBTITLE Hydrologic Modeling System HEC-HMS Quick Start Guide	5. FUNDING NUMBERS
6. AUTHOR(S)	

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) U.S. Army Corps of Engineers Hydrologic Engineering Center, HEC 609 Second St. Davis, CA 95616-4687	8. PERFORMING ORGANIZATION REPORT NUMBER

9. SPONSORING / MONITORING AGENCY NAME(S) AND ADDRESS(ES) HQ U.S. Army Corps of Engineers 441 G St., NW Washington, DC 20314-1000	10. SPONSORING / MONITORING AGENCY REPORT NUMBER

11. SUPPLEMENTARY NOTES

12A. DISTRIBUTION / AVAILABILITY STATEMENT Distribution is unlimited.	12B. DISTRIBUTION CODE

13. ABSTRACT *(Maximum 200 words)*

The Hydrologic Modeling System (HEC-HMS) is designed to simulate the precipitation-runoff processes of dendritic watershed systems. It supersedes HEC-1 and provides a similar variety of options but represents a significant advancement in terms of both computer science and hydrologic engineering. In addition to unit hydrograph and hydrologic routing options, capabilities include a linear quasi-distributed runoff transform (ModClark) for use with gridded precipitation, continuous simulation with either a one-layer or more complex five-layer soil moisture method, gridded and subbasin average Priestly-Taylor evapotranspiration, a gridded and elevation-band temperature-index snowmelt method, multiple outlets and spillways for reservoirs, and an automatic depth-area reduction analysis tool for frequency storms.

The program features a completely integrated work environment including a database, data entry utilities, computation engine, and results reporting tools. The user interface allows the user seamless movement between the different parts of the program. Simulation results are stored in the Data Storage System HEC-DSS and can be used in conjunction with other software for studies of water availability, urban drainage, flow forecasting, future urbanization impact, reservoir spillway design, flood damage reduction, floodplain regulation, and systems operation.

14. SUBJECT TERMS Hydrology, watershed, precipitation runoff, river routing, flood control, water supply, computer simulation.	15. NUMBER OF PAGES 44
	16. PRICE CODE

17. SECURITY CLASSIFICATION OF REPORT Unclassified	18. SECURITY CLASSIFICATION OF THIS PAGE Unclassified	19. SECURITY CLASSIFICATION OF ABSTRACT Unclassified	20. LIMITATION OF ABSTRACT Unlimited

Hydrologic Modeling System
HEC-HMS

Quick Start Guide DRAFT

Version 3.0
June 2005

C H A P T E R   1

Introduction

This document was developed using the Hydrologic Engineering Center's Hydrologic Modeling System (HEC-HMS) version 3.0.

HEC-HMS is designed to simulate the precipitation-runoff processes of dendritic watershed systems. It is designed to be applicable in a wide range of geographic areas for solving a broad range of problems. This includes large river basin water supply and flood hydrology to small urban or natural watershed runoff. Hydrographs produced by the program can be used directly or in conjunction with other software for studies of water availability, urban drainage, flow forecasting, future urbanization impact, reservoir spillway design, flood damage reduction, floodplain regulation, wetlands hydrology, and systems operation.

This document was written as a brief introduction to the program and will be more beneficial to users with experience using previous versions of HEC-HMS. For less experienced users, a more comprehensive description and application of the program can be found in the *HMS Technical Reference Manual* and the *HMS Applications Guide*. These documents will be updated to reflect changes in the program after the release of HMS version 3.0 beta. The Quick Start Guide is divided into the following three chapters: Chapter 1 provides a description of program components and the user interface, Chapter 2 lists and describes steps required to develop a hydrologic model and obtain results, and Chapter 3 contains an example application following the steps outlined in Chapter 2. An appendix is included to provide a description of the optimization and the depth-area analysis features.

HMS Model Components

HEC-HMS model components are used to simulate the hydrologic response in a watershed. HMS model components include basin models, meteorologic models, control specifications, and input data. A simulation calculates the precipitation-runoff response in the basin model given input from the meteorologic model. The control specifications define the time period and time step of the simulation run. Input data components, such as time-series data, paired data, and gridded data are often required as parameter or boundary conditions in basin and meteorologic models.

Basin Model Component

The basin model represents the physical watershed. The user develops a basin model by adding and connecting hydrologic elements. Hydrologic elements use mathematical models to describe physical processes in the watershed. Table 1 provides a list and description of available hydrologic elements.

Table 1.   Hydrologic Element Description.

Hydrologic Element	Description
Subbasin	The subbasin element is used to represent the physical watershed. Given precipitation, outflow from the subbasin element is calculated by subtracting precipitation losses, transforming excess precipitation to stream flow at the subbasin outlet, and adding baseflow.
Reach	The reach element is used to convey stream flow downstream in the basin model. Inflow into the reach element can come from one or many upstream hydrologic elements. Outflow from the reach is calculated by accounting for translation and attenuation of the inflow hydrograph.
Junction	The junction element is used to combine stream flow from hydrologic elements located upstream of the junction element. Inflow into the junction element can come from one or many upstream elements. Outflow is simply calculated by summing all inflows and assuming no storage at the junction.
Source	The source element is used to introduce flow into the basin model. The source element has no inflow. Outflow from the source element is defined by the user.
Sink	The sink element is used to represent the outlet of the physical watershed. Inflow into the sink element can come from one or many upstream hydrologic elements. There is no outflow from the sink element.
Reservoir	The reservoir element is used to model the detention and attenuation of a hydrograph caused by a reservoir or detention pond. Inflow into the reservoir element can come from one or many upstream hydrologic elements. Outflow from the reservoir element can be calculated two ways. The user can enter a storage-outflow, elevation-storage-outflow, or elevation-area-outflow relationship, or the user can enter an elevation-storage or elevation-area relationship and define one or more outlet structures.
Diversion	The diversion element is used for modeling stream flow leaving the main channel. Inflow into the diversion element can come from one or many upstream hydrologic elements. Outflow from the diversion element consists of diverted flow and non-diverted flow. Diverted flow is calculated using input from the user. Both diverted and non-diverted flows can be connected to hydrologic elements downstream of the diversion element.

In the case of the subbasin element, many mathematical models are available for determining precipitation losses, transforming excess precipitation to stream flow at the subbasin outlet, and adding baseflow. In this document the different mathematical models will be referred to as methods. Table 2 lists the methods available for subbasin and river reach elements.

Table 2.   Subbasin and Reach calculation methods.

Hydrologic Element	Calculation Type	Method
Subbasin	Runoff-volume	Deficit and constant rate (DC)
		Exponential
		Green and Ampt
		Gridded DC
		Gridded SCS CN
		Gridded SMA
		Initial and constant rate
		SCS curve number (CN)
		Soil moisture accounting (SMA)
	Direct-runoff	Clark's UH
		Kinematic wave
		ModClark
		SCS UH
		Snyder's UH
		User-specified s-graph
		User-specified unit hydrograph (UH)
	Baseflow	Bounded recession
		Constant monthly
		Linear reservoir
		Recession
Reach	Routing	Kinematic wave
		Lag
		Modified Puls
		Muskingum
		Muskingum-Cunge

Meteorologic Model Component

The meteorologic model calculates the precipitation input required by a subbasin element. The meteorologic model can utilize both point and gridded precipitation and has the capability to model frozen and liquid precipitation along with evapotranspiration. The newly added snowmelt method uses a temperature index algorithm to calculate the accumulation and melt of the snow pack. The evapotranspiration methods include the constant monthly method and the new Priestly Taylor and gridded Priestly Taylor methods. An evapotranspiration method is only required when simulating the continuous or long term hydrologic response in a watershed. A brief description of the methods available for calculating basin average precipitation or grid cell precipitation is included in Table 3.

Table 3.   Description of meteorologic model methods.

Precipitation Methods	Description
Frequency Storm	This method is used to develop a precipitation event where precipitation depths for various durations within the storm have a consistent exceedance probability.
Gage Weights	This method applies user specified weights to user defined precipitation gages.
Gridded Precipitation	This method allows the use of gridded precipitation products, such as RADAR.
Inverse Distance	This method calculates subbasin average precipitation by applying an inverse distance squared weighting to user defined precipitation gages.
SCS Storm	This method applies a user specified SCS time distribution to a 24-hour total storm depth.
Specified Hyetograph	This method applies a user defined hyetograph to a specified subbasin element.
Standard Project Storm	This method applies a time distribution to an index precipitation depth.

Control Specifications Component

The control specifications set the time span of a simulation run. Information in the control specifications includes a starting date and time, ending date and time, and computation time step.

Input Data Components

Time-series data, paired data, and gridded data are often required as parameter or boundary conditions in basin and meteorologic modeling methods. A complete list of input data is included in Table 4. Input data can be entered manually or referenced to an existing record in a HEC-DSS file. All gridded data must be referenced to an existing HEC-DSS record. Refer to _HEC-DSSVue User's Manual_ (USACE, 2003) for a description of HEC-DSS.

Table 4.   _Input data components._

Time-Series Data	Paired Data	Gridded Data
Precipitation	Storage-outflow	Precipitation
Discharge	Elevation-storage	Temperature
Temperature	Elevation-area	Solar radiation
Solar radiation	Elevation-discharge	Crop coefficient
Crop Coefficient	Inflow-diversion	Storage capacity
	Cross sections	Percolation rate
	Unit hydrograph curves	Storage coefficients
	Percentage curves	Moisture deficit
	ATI-meltrate functions	Impervious area
	ATI-coldrate functions	SCS curve number
	Groundmelt patterns	Elevation
	Evaporation patterns	Cold content
	Meltrate patterns	Cold content ATI
		Meltrate ATI
		Liquid water content
		Snow water equivalent

User Interface

The user interface consists of a menu bar, tool bar, and four main panes. Starting from the upper left pane in Figure 1 and moving counter-clockwise, these panes will be referred to as the Watershed Explorer, the Component Editor, the Message Log, and the Desktop.
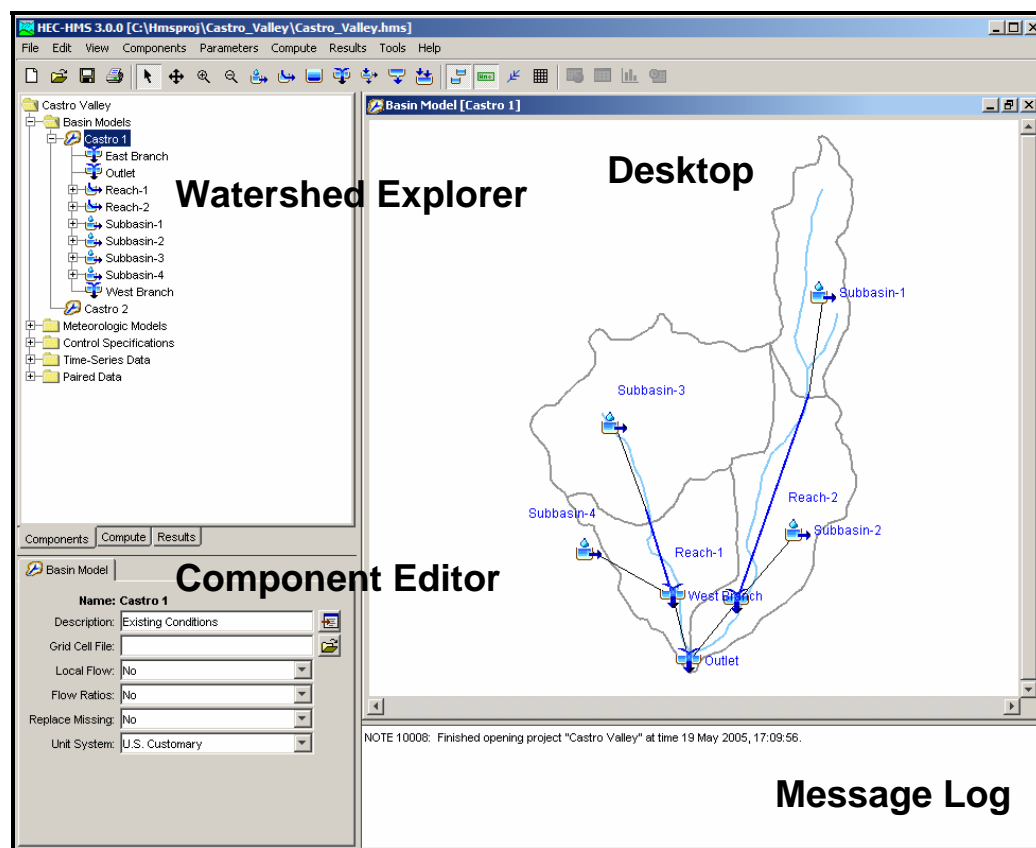
Figure 1.   HMS-HMS user interface.

Watershed Explorer

The watershed explorer was developed to provide quick access to all components in an HEC-HMS project. For example, the user can easily navigate from a basin model to a precipitation gage and then to a meteorologic model without using menu options or opening additional windows. The watershed explorer is divided into three parts: Components, Compute, and Results. The arrow in Figure 2 points to the "Components" tab of the watershed explorer. The hierarchal structure of model components, such as basin models, meteorologic models, etc., is available from the "Components" tab of the watershed explorer. The watershed explorer organizes model components into individual folders. When a component is selected, the watershed explorer expands to show sub-components. For example, when a basin model is selected the watershed explorer will expand to show all hydrologic elements in the basin model. Notice in Figure 2 that the Castro 1 basin model is selected and the watershed explorer is expanded to show all hydrologic elements in the basin model. The plus/minus sign beside model components and sub-components can be used to expand/collapse the watershed explorer. The Compute watershed explorer, accessed from the "Compute" tab, contains all project simulation runs, optimization trials, and analyses. Model results

are available from the "Results" tab of watershed explorer. Results from different simulations can be compared in the same graph or table.
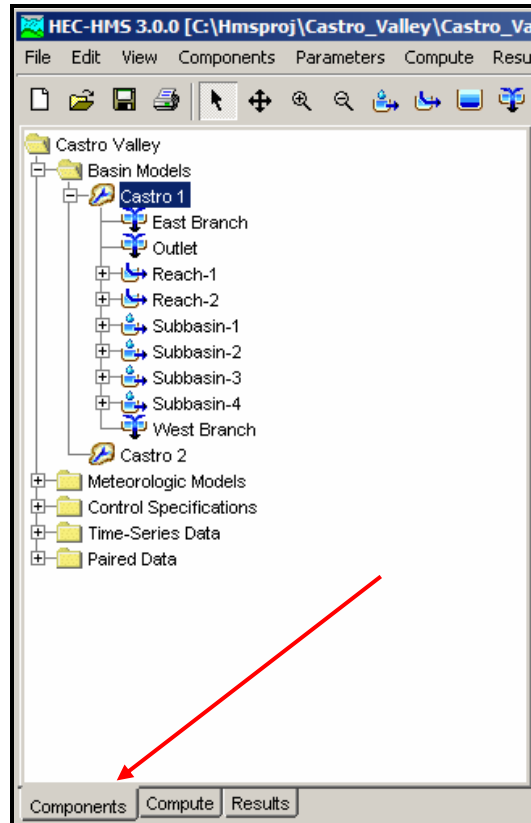
Figure 2.   Watershed explorer.

Component Editor

When a component or sub-component in the watershed explorer is active (simply use the mouse and click on the component name in the watershed explorer), a specific component editor will open. All data required by model components are entered in the component editors. For example, loss parameter data for a subbasin element is entered in the subbasin component editor. The component editor for the `Castro 1` basin model is shown in Figure 3.
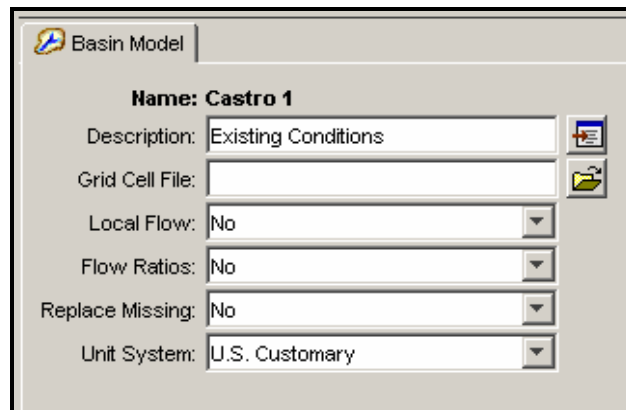
Figure 3.   Component editor.

Message Log

Notes, warnings, and errors are shown in the message log (Figure 4). These messages are useful for identifying why a simulation run failed or why a requested action, like opening a project, was not completed. A comprehensive list and description of messages will be provided in the new *HMS User's Manual*.

NOTE 10008: Finished opening project "Castro Valley" at time 19 May 2005, 17:09:56.
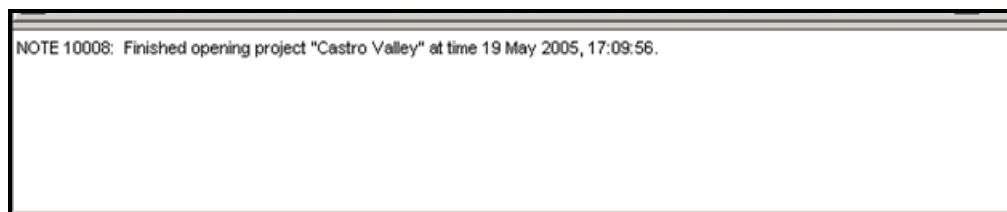
Figure 4.   Message log.

Desktop

The desktop holds a variety of windows including summary tables, time-series tables, graphs, global editors, and the basin model map. The basin model map is used to develop a basin model. Elements (subbasin, river reach, reservoir, etc.) are added from the toolbar and connected to represent the physical drainage network of the study area. Background maps can be imported to help visualize the watershed. The Castro 1 basin model map is shown in Figure 5.
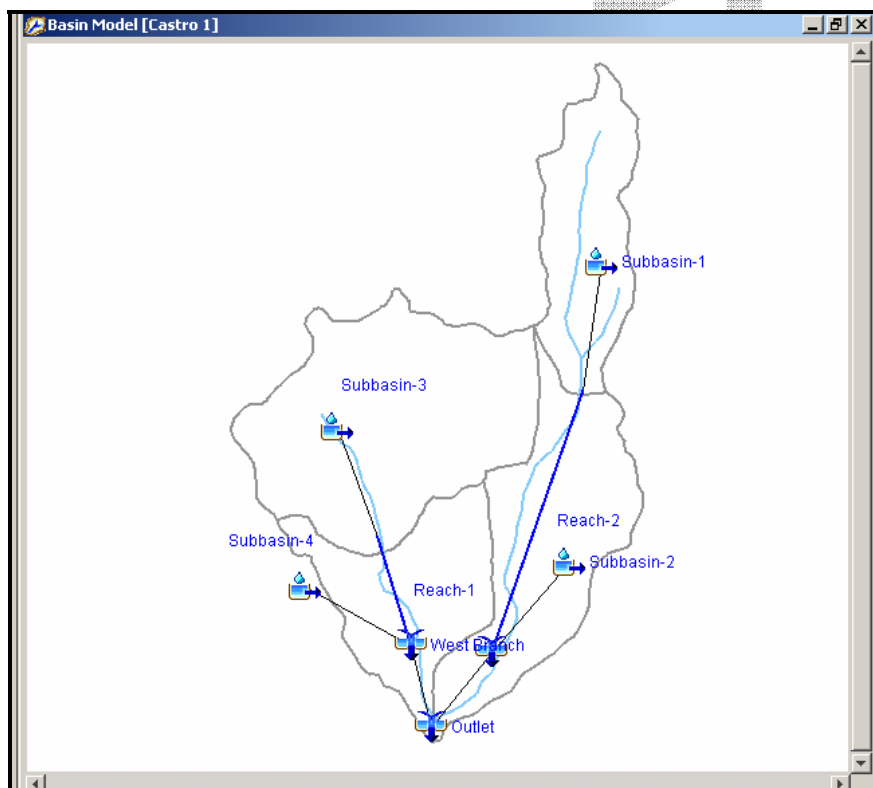
Figure 5.   Basin model map opened in the Desktop.

Documentation Conventions

The following conventions are used throughout the Quick Start Guide to describe the HMS user interface.

- Screen titles are shown in *italics*.

- Menu names, menu items, and button names are shown in **bold**.

- Menus are separated from submenus with the right arrow $\Rightarrow$.

- Data to be typed into an input field on a screen is shown in the `courier` font.

- A column heading, tab name, or field title is shown in "double quotes".

References

USACE (2003). *HEC-DSSVue User's Manual*. Hydrologic Engineering Center, Davis, CA.

C H A P T E R   2

Developing an HMS Project

To develop a hydrologic model, the user must complete the following steps:

- Create a new project.

- Input time series, paired, and gridded data needed by the basin or meteorologic model.

- Define the physical characteristics of the watershed by creating and editing a basin model.

- Select a method for calculating subbasin precipitation and enter required information. Evapotranspiration and snow melt information are also entered at this step if required.

- Define the control specifications.

- Combine a basin model, meteorologic model, and control specifications to create a simulation.

- View the results and modify the basin model, meteorologic model, or control specifications as needed.

Create a New Project

Create a new project by selecting **File** ⇒ **New** from the menu bar (Figure 6). Enter a project "Name", select a "Location" for storing project files, and enter a project "Description" in the *Create a New Project* screen (Figure 7). A new folder with the same name as the project name is created in the selected directory. This folder will store all files created for this project. External HEC-DSS files, ModClark files, and background map files do not have to be stored in the project folder. A new project can also be created by selecting the **Create a New Project** button on the tool bar.

Options for managing a project are available from the **File** menu option. These options include **Open**, **Save**, **Save As**, **Delete**, and **Rename**. The tool bar contains buttons to **Open** a project and **Save** the current project.

_____
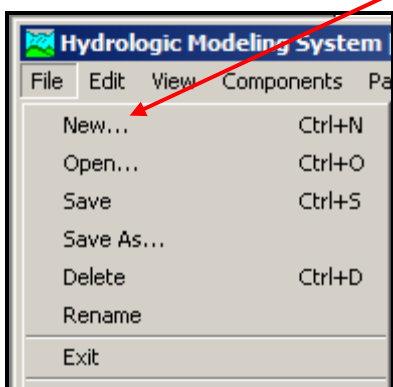
Figure 6.   Create a new project.
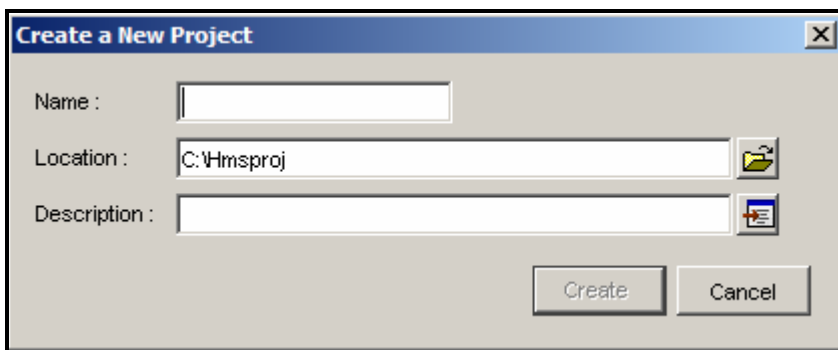
Figure 7.   Enter a project name, location of project files, and project description.

Input Data

Time series data, paired data, and gridded data are created by selecting the **Components** menu option and then selecting either the **Time-Series Gage Manager**, **Paired Data Manager**, or **Grid Manager** (Figure 8) input data type. A manager window opens that allows the user to select the gage, paired data, or grid type. Buttons on the right side of the manager provide options to create a **New**, **Copy**, **Rename**, or **Delete** the data type. Figure 9 shows the *Paired Data Manager* (The `Storage-Outflow` data type is selected). Once a new input data type has been created, required information can be entered in the component editor. Input data can be entered manually or referenced to an existing record in a HEC-DSS file.
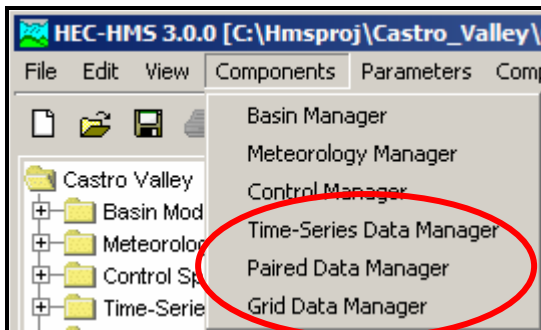
Figure 8.   Input data.
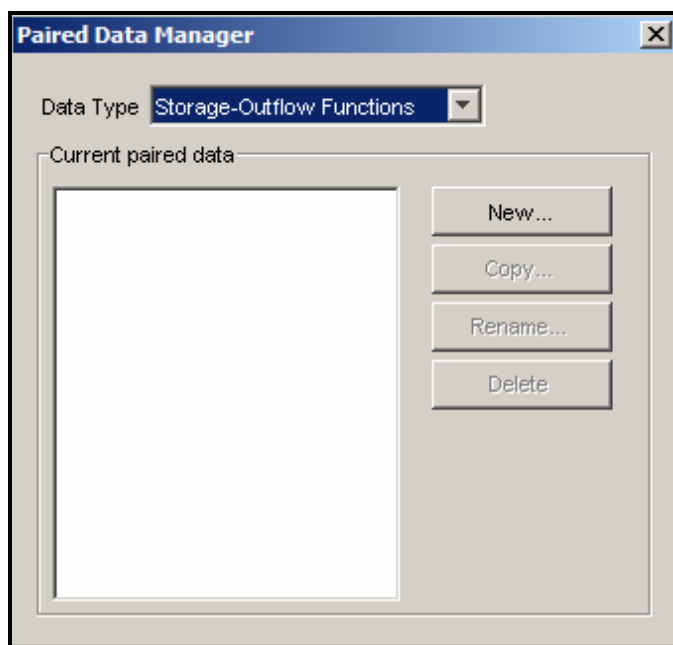
Figure 9.   Paired data manager.

Figure 10 shows the component editor for a Storage-Discharge table. The table can be renamed in the watershed explorer or in the paired data manager. The "Data Source" options are `Manual Entry` and `Data Storage System (HEC-DSS)`. If `Data Storage System (HEC-DSS)` is selected, the user is required to select a HEC-DSS file and a pathname. If `Manual Entry` is selected, the user must click the "Table" tab and manually enter the storage-discharge values.

A time window is required for time-series data before the data can be entered or viewed. A default time window is provided when a time-series gage is added to the project. To add an additional time window, click the right mouse button when the mouse pointer is on the gage's name in the watershed explorer and select the **Create Time Window** option (Figure 11).
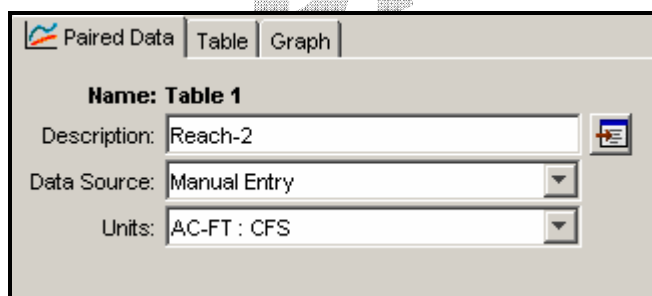
Figure 10.   Component editor for a storage-discharge table.
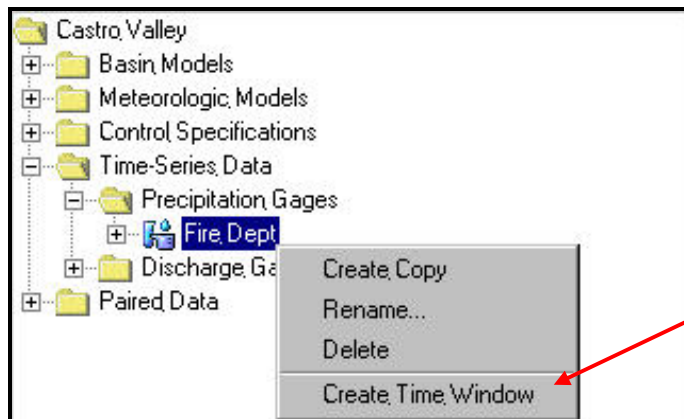
Figure 11.   Create a time window for a time-series gage.

Create a Basin Model

A new basin model can be added to a project by selecting the **Component ⇒ Basin Manager** menu option (Figure 12). Click the **New** button in the *Basin Model Manager* window. Enter a "Name" and "Description" in the *Create A New Basin Model* window and click the **Create** button (Figure 13). An existing basin model can be added to an HMS project by selecting the **Tools ⇒ Import ⇒ Basin Model** menu option.

Once a new basin model has been added, hydrologic elements can be added and connected in the basin model map to reflect the drainage of the real world watershed. Hydrologic elements are added by selecting one of the element tools on the tool bar (Figure 14) and clicking the left mouse button in the basin model map. To connect an upstream element to a downstream element, click the right mouse button when the mouse pointer is on top of the element icon and select the **Connect Downstream** option. Then place the pointer on top of the desired downstream element and click the left mouse button.

Copy, **Rename**, or **Delete** the basin model by clicking the right mouse button when the pointer is on top of the basin model name in the watershed explorer. These options are also available from the basin model manager. **Copy**, **Rename**, or **Delete** hydrologic elements from the watershed explorer. The **Copy** and **Delete** options are also available by clicking the right mouse button when the mouse pointer is located on top of the hydrologic element icon in the basin model map.

Basin model and hydrologic element parameter data are entered in the component editor. Select a basin model name or hydrologic elements name in the watershed explorer to open the component editor. Hydrologic element component editors can also be opened by selecting the element icon in the basin model map. Figure 15 shows a component editor for a subbasin element. Notice the five tabs labeled "Subbasin," "Loss," "Transform," "Baseflow," and "Options."
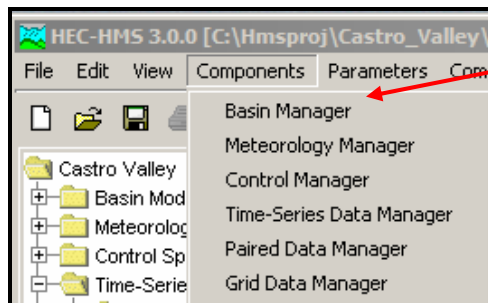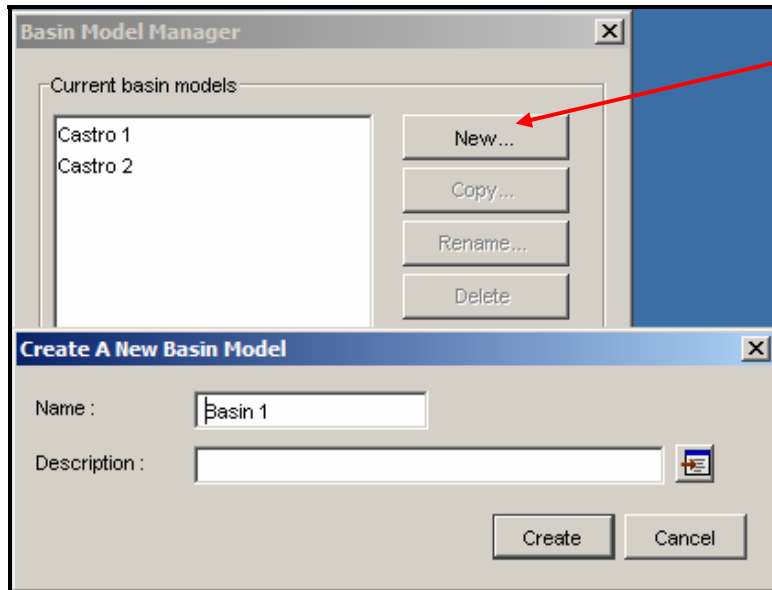
Figure 12.   Open the basin manager.

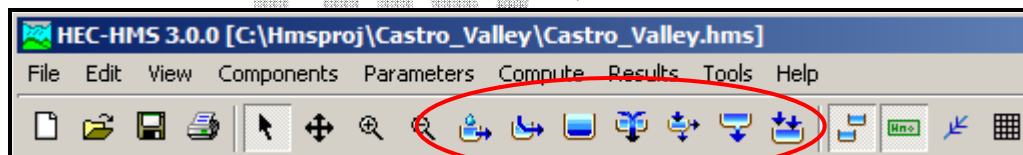*Figure 13.   Create a new basin model.*

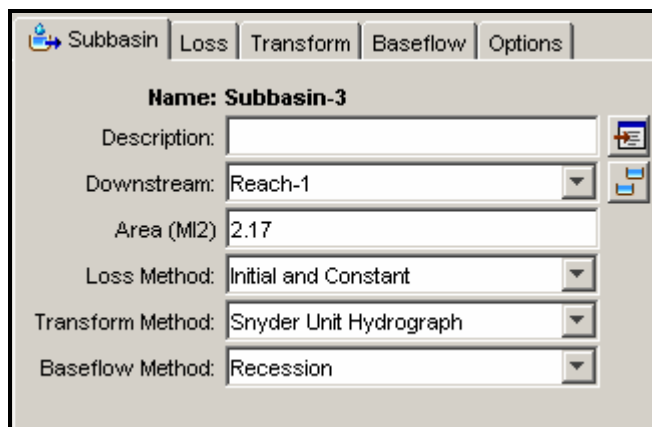*Figure 14.   Hydrologic element tools.*

Figure 15.   Component editor for a subbasin element.

Create a Meteorologic Model

A meteorologic model is added to a project in the same manner as the basin model. Select the **Component ⇒ Meteorology Manager** menu option. Click the **New** button in the *Meteorologic Manager* window and enter a "Name" and "Description" in the *Create A New Meteorologic Model* window. To import an existing meteorologic model, select the **Tools ⇒ Import ⇒ Meteorologic Model** menu option. The meteorologic model can be renamed in the watershed explorer or using the meteorologic manager. Figure 16 shows the component editor for a meteorologic model.

One step in developing a meteorologic model is to define which basin models require precipitation from the meteorologic model. Select the "Basins" tab in the meteorologic model component editor. Change the "Include Subbasins" option to "Yes" for all basin models requiring precipitation from the selected meteorologic model (Figure 17). All subbasin elements contained in the selected basin model(s) will be added to the meteorologic model. Once added, parameters for the precipitation, evapotranspiration, and snowmelt methods can be defined for each subbasin element using the component editor.

Figure 16.   Meteorologic model component editor.

Figure 17.   Adding subbasin elements to a meteorologic model.

Define Control Specifications

A control specifications is added to a project by selecting the **Component** ⟹ **Control Manager** menu option. Click the **New** button in the *Control Specifications Manager* window and enter a "Name" and "Description" in the *Create a New Control Specifications* window. The component editor (Figure 18) for a control specifications requires, a start date and time, an end date and time, and a time step. Start and end dates must be entered using the "ddMMMYYYY" format, where "d" represents the day, "M" represents the month, and "Y" represents the year. Time is entered using the 24 hour format. Start and end times must be entered using the "HH:mm" format, where "H" represents the hour and "m" represents the minute. A colon is required when separating hour and minute. The time step is selected from an available interval list containing time steps from 1 minute to 24 hours. Calculations for most methods are performed using the specified time step; output is always reported in the specified time step.

Figure 18.   Control specifications.

Create and Compute a Simulation Run

A simulation run is created by selecting the **Compute** ⟹ **Run Manager** menu option. Click the **New** button in the *Simulation Run Manager* window. The simulation run manager also allows the user to **Copy**, **Rename**, and **Delete** an existing simulation run. After clicking the **New** button, a wizard opens to step the user through the process of creating a simulation run. First, a name must be entered for the simulation run, then a basin model, a meteorologic model, and a control specifications must be selected. The new simulation run is added to the "Compute" tab of the watershed explorer (Figure 19). A simulation run can also be created by selecting the **Compute** ⟹ **Create Simulation Run** menu option. In the simulation run component editor, the user can enter a "Description" and change the basin model, meteorologic model, and control specifications from drop-down lists. The simulation run can be renamed in the watershed explorer or from the simulation run manager. Click the right mouse button when the pointer is on top of the simulation run's name in the watershed explorer and select the **Rename** option. Other options available when clicking the right mouse button include **Compute**, **Create Copy**, and **Delete**. A simulation run can also be computed by selecting the **Compute** ⟹ **Select Run** menu option and choosing the desired simulation run (Figure 20). Then, select the **Compute** menu option and click the **Compute Run** option at the bottom of the menu (Figure 21).
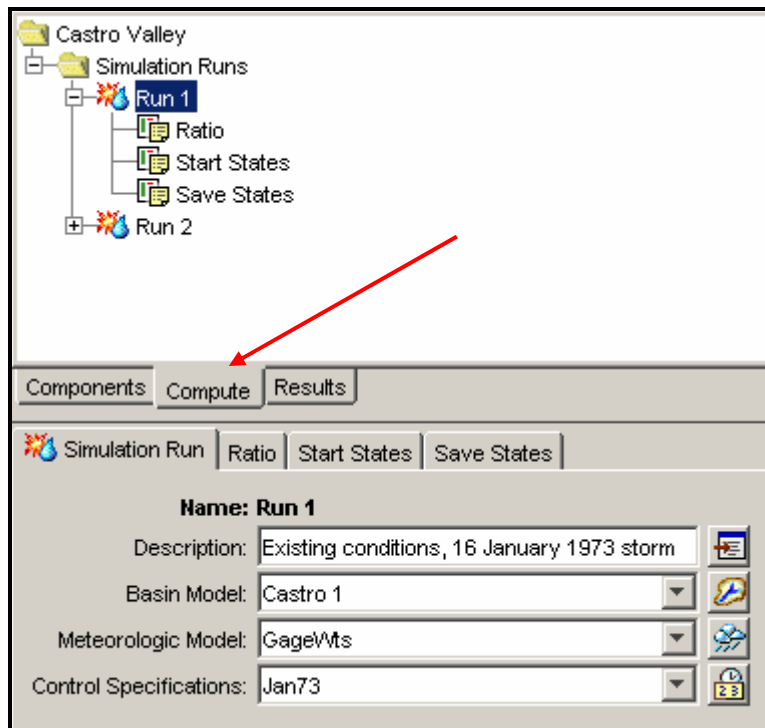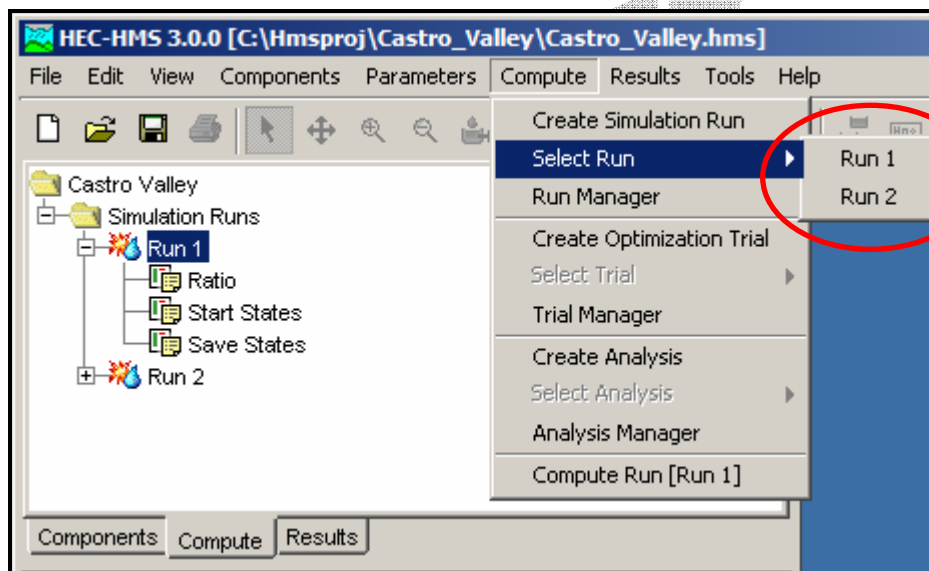
Figure 19.   Simulation run.
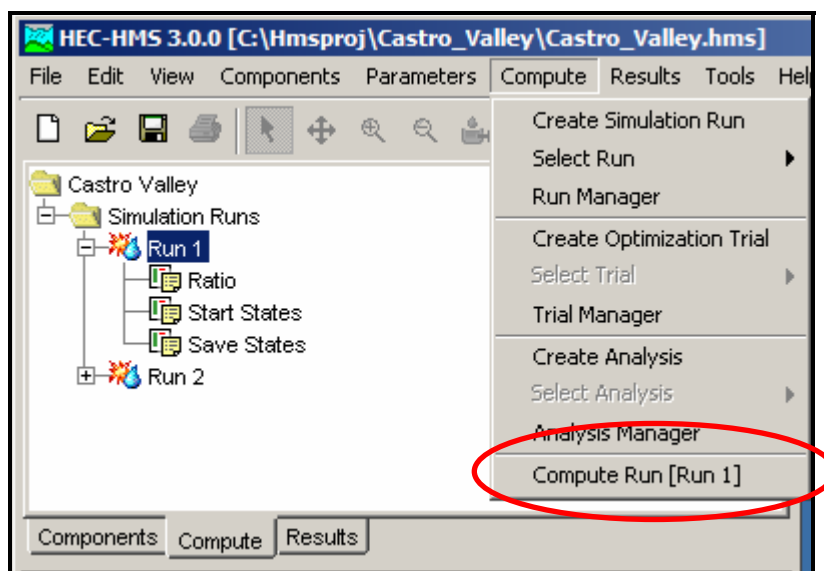

Figure 20.   Selecting a simulation run.

Figure 21.   Computing the selected simulation run.

View Model Results

Graphical and tabular results are available after a simulation run, an optimization trial, and an analysis have been computed (refer to the appendix for a description of optimization trials and analyses). Results can be accessed from the watershed explorer or the basin map. Results for an optimization trial or analysis can only be accessed from the watershed explorer. Results are available as long as no edits were made to model components (subbasin parameters, time-series data, etc.) after the simulation run, optimization trial, or analysis were computed. If edits were made, the simulation run, optimization trial, or analysis must be re-computed.

Select the "Results" tab in the watershed explorer to view a list of simulation runs, optimization trials, and analyses (Figure 22). Click the box next to the name of the simulation run, optimization trial, or analysis to expand the watershed explorer. Click the box next to a hydrologic element's name to expand the watershed explorer even more to show available results for the hydrologic element. When a times series result is selected in the watershed explorer, a preview graph opens in the component editor. Figure 22 shows a times series graph for a subbasin element (Subbasin-1). Multiple time series records can be added to the same graph by holding the Control key and clicking other time series results in the watershed explorer. Time series results from different basin model elements and from different simulation runs and optimization trials can be added to the same graph for comparison (Figure 23). A copy of the preview graph will be

added to the desktop by clicking the graph button on the toolbar (Figure 23).

Results can also be accessed from the basin model map. After a simulation run computes, click the right mouse button when the mouse pointer is located on top of a basin model element in the basin map. Select the **View Results** option and choose **Graph**, **Summary Table**, or **Time-Series Table** (Figure 24). Results are also available from the toolbar. Select a basin model element in the basin map or watershed explorer to make it active. Then choose one of the table, graph, or times-series table buttons on the toolbar.
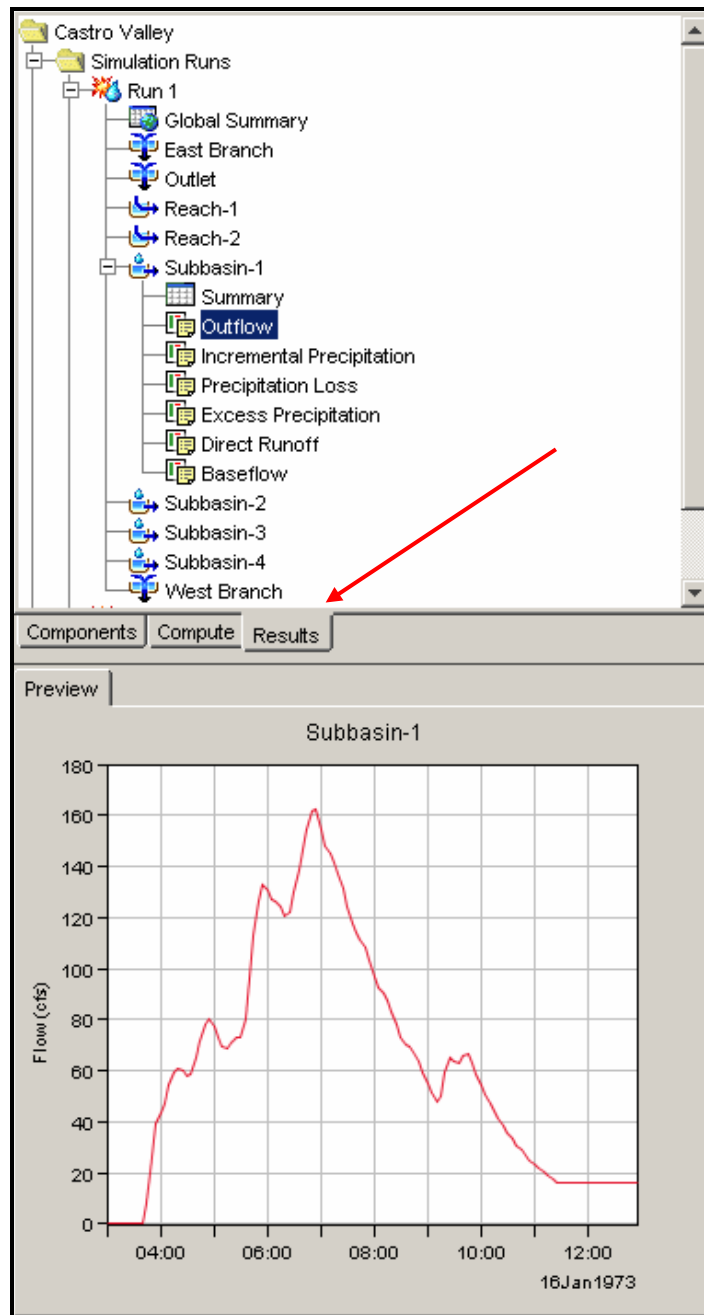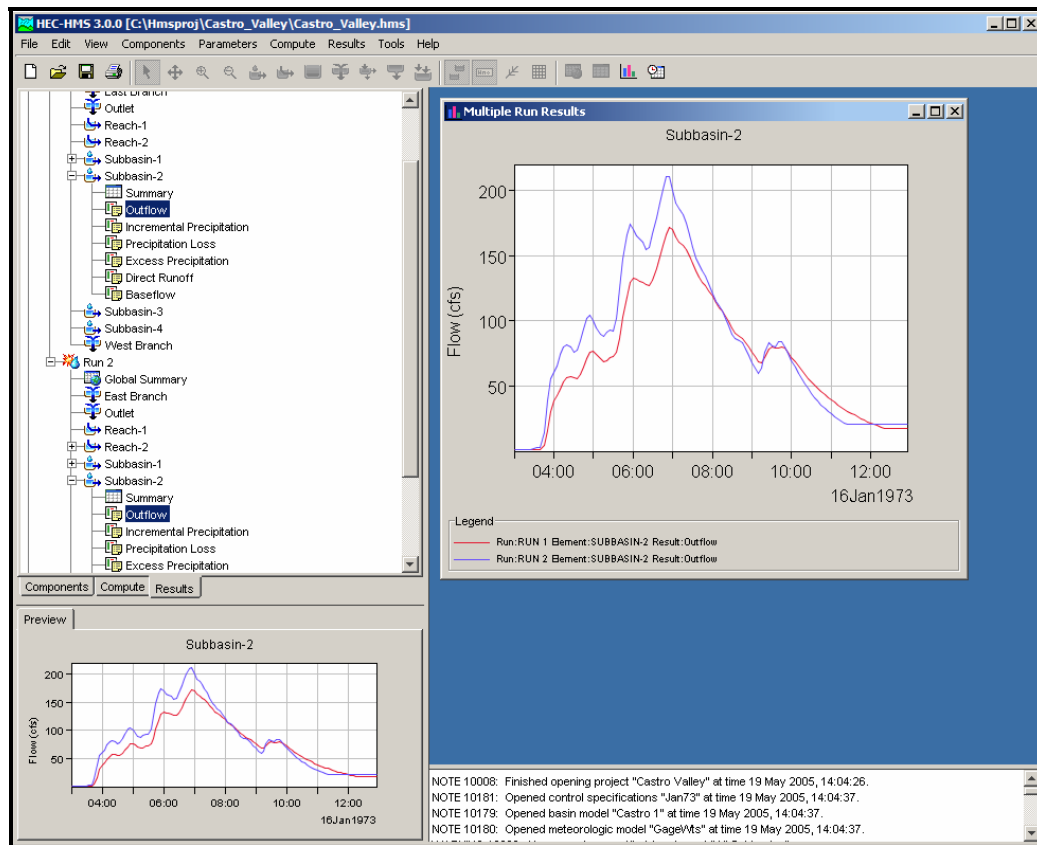
Figure 22.   Viewing results.

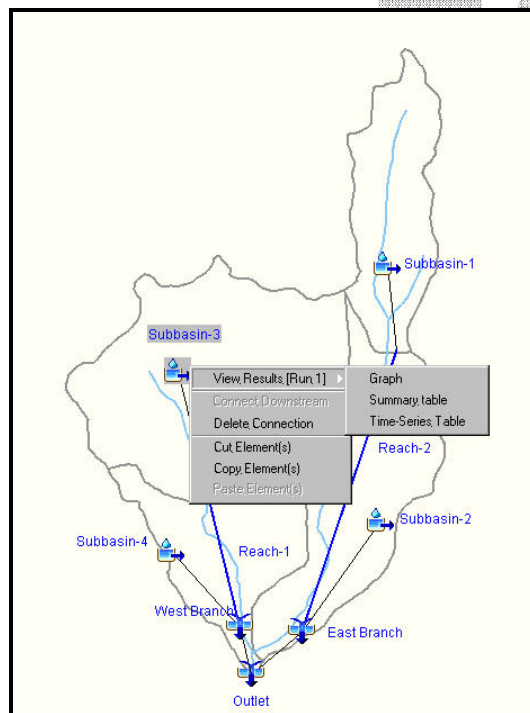*Figure 23.   Comparing results from different simulation runs.*

Figure 24.   Viewing results from the basin model map.

C H A P T E R   3

Example

This chapter illustrates the steps necessary to create a precipitation-runoff model with an example.

Problem Statement

This example uses data from the 5.51 square mile Castro Valley watershed located in northern California. The watershed contains four major catchments (Figure 25). Precipitation data for a storm that occurred on January 16, 1973, is available for three gages in the watershed: Proctor School, Sidney School, and Fire Department. The goal of the example is to estimate the affect of proposed future urbanization on the hydrologic response.
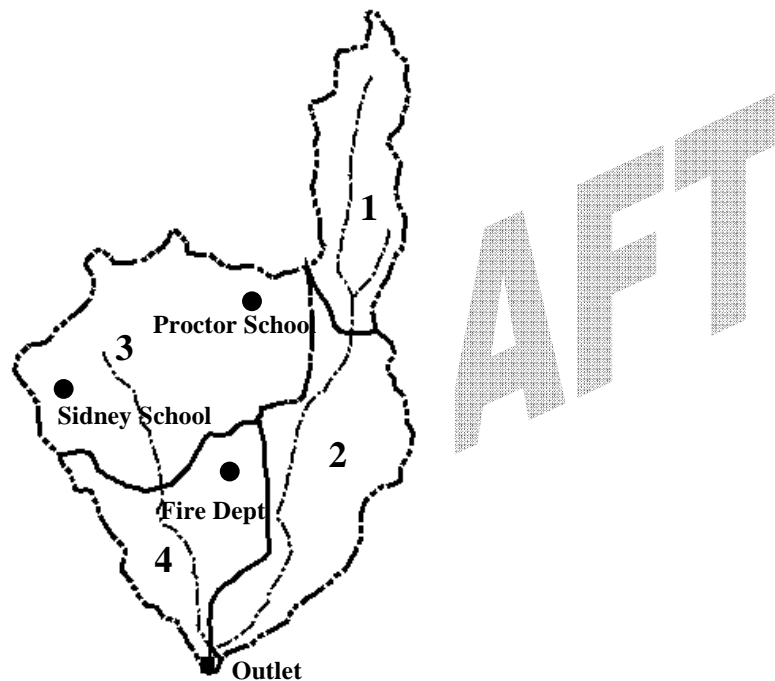
Figure 25.   Castro Valley Creek watershed.

Application of the program will require creating a new project and entering gage data. A basin model using the initial constant loss, Snyder unit hydrograph transform, and recession baseflow methods will be created from the parameter data shown in Tables 5 - 8.

Table 5. *Subbasin initial and constant loss method and Snyder transform method data.*

Subbasin	Loss Parameters			Transform Parameters	
ID	Initial	Constant	Impervious	t_p	C_p
	in	*in/hr*	*%*	*hr*	
1	0.02	0.14	2	0.20	0.16
2	0.02	0.14	8	0.28	0.16
3	0.02	0.14	10	0.20	0.16
4	0.02	0.14	15	0.17	0.16

Table 6. *Subbasin area and baseflow data.*

Subbasin Parameters		Baseflow Parameters		
ID	Area	Initial Flow	Threshold	Recession
	sq-mi	*cfs/sq-mi*	*ratio-to-peak*	*constant*
1	0.86	0.54	0.1	0.79
2	1.52	0.54	0.1	0.79
3	2.17	0.54	0.1	0.79
4	0.96	0.54	0.1	0.79

Table 7. *Routing criteria for reaches.*

ID	From	To	Method	Sub-reaches	Parameters
Reach-2	Subbasin-1	East Branch	Muskingum	7	K = 0.6 hr, x = 0.2
Reach-1	Subbasin-3	West Branch	Modified Puls	4	in = out, Table 8

Table 8. *Storage-discharge data for Reach 2.*

Storage	Outflow
ac-ft	*cfs*
0	0
0.2	2
0.5	10
0.8	20
1.0	30
1.5	50
2.7	80
4.5	120
750	1,500
5,000	3,000

_____

A meteorologic model will have to be created for the precipitation data. Thiessen polygon weights (Table 9) will be used for a user gage weighting precipitation method. Total rainfall measured by the Proctor School and Sidney School gages was 1.92 and 1.37 inches, respectively. Storm rainfall is to be distributed in time using the temporal pattern of incremental precipitation from the Fire Department gage. The Fire Department gage data has been stored in the HEC-DSS file `CASTRO.DSS` with the following pathname: `/CASTRO VALLEY/FIRE DEPT./PRECIP-INC/16JAN1973/10MIN/OBS/`. The DSS file is part of the `Castro` example project (Look in the "hmsproj\Castro" folder, or wherever the project data was installed).

Table 9.   Precipitation gage weights.

Subbasin	Proctor School	Fire Dept.	Sidney School
1	1.00	0.00	0.00
2	0.20	0.80	0.00
3	0.33	0.33	0.33
4	0.00	0.80	0.20

A simulation run for pre-development conditions will be created and computed to determine the existing conditions rainfall-runoff response. Finally, future urbanization will be modeled and results compared to the existing conditions.

Create the Project

Begin by starting HEC-HMS and creating a new project. Select the **File ⇒ New** menu item. Enter `Castro Valley` for the project "Name" and `Castro Valley Urban Study` for the "Description" (Figure 26). Project files will be stored in a directory called `Castro_Valley`, a subdirectory of the "hmsproj" directory created during program installation. Click the **Create** button to create the project.
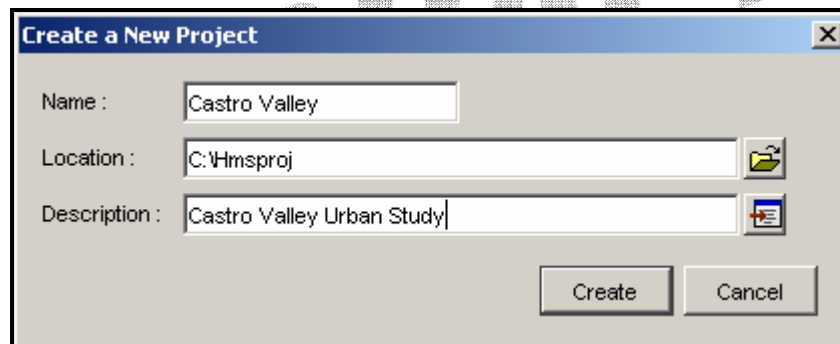
Figure 26.   Enter the name and description of the new project.

Set the project attributes before creating gages or model components (Figure 27). Select the **Tools ⇒ Options** menu item. Click the "Defaults" tab. Set "Unit system" to `U.S. Customary`, "Loss" to `Initial and Constant`, "Transform" to `Snyder Unit Hydrograph`, "Baseflow" to `Recession`, "Routing Method" to `Muskingum`, "Precipitation" to `Gage Weights`, "Evapotranspiration" to `None`, and "Snowmelt" to `None`. Click the **OK** button to save and close the *Options* window.
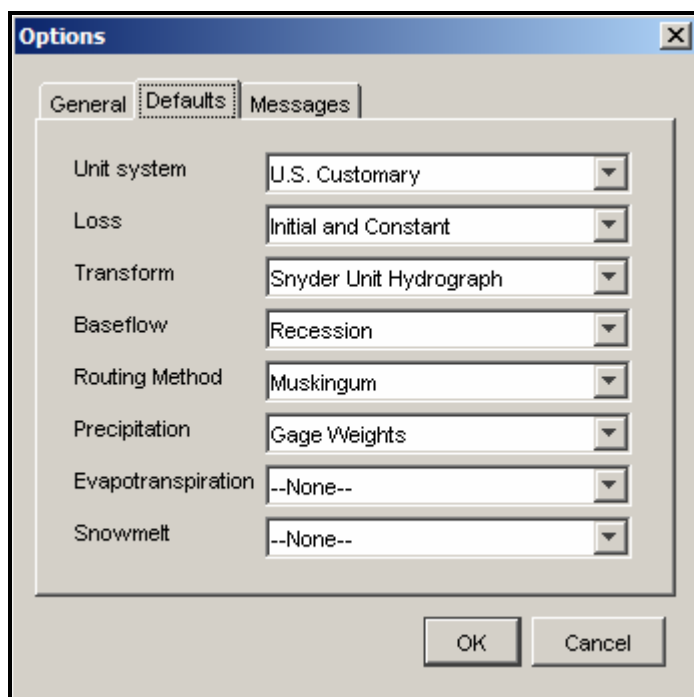
Figure 27.   Setting the project attributes.

Input Data

Create a precipitation gage for the Fire Department data. Select the **Component** ⇒ **Time-Series Data Manager** menu item. Make sure the "Gage Type" is set to `Precipitation Gages`. Click the **New** button in the *Gage Manager* window. In the *Create A New Precipitation Gage* window enter `Fire Dept` for the "Name" and `Castro Valley Fire Department` for the "Description". Click the **Create** button to add the precipitation gage to the project. The component editor opens automatically and the `Fire Dept` precipitation gage is added to the **Precipitation Gages** folder under the **Time-Series Data** component in the watershed explorer. The precipitation gage component editor contains four tabs: "Time-Series Gage", "Time Window", "Table", and "Graph." Select the "Time-Series Gage" editor and select the `Data Storage System`

`(HEC-DSS)` "Data Source" option. Click the **DSS Filename** button and locate the

`CASTRO.dss` file. Click the **DSS Pathname** button to view a list of records in the DSS file. Select the `/CASTRO VALLEY/FIRE DEPT./PRECIP-INC/16JAN1973/10MIN/OBS/` pathname (Figure 28).

To view a time series table and graph of precipitation data, open the "Time Window" editor. Enter a "Start Date" and "End Date" of `16 January 1973`, a "Start Time" of `03:10`, and an "End Time" of `09:50`. Click on the "Table" tab to view a table and click the "Graph" tab to view a graph of the `Fire Dept` precipitation data.

_____

Figure 28.   Component editor for the Fire Department precipitation gage.

Create a discharge gage for the observed hydrograph at the watershed outlet using the same procedure for the precipitation gage. Select the **Component ⇒ Time-Series Data Manager** menu item. Make sure the "Gage Type" is set to Discharge Gages. Click the **New** button in the *Gage Manager* window. In the *Create A New Discharge Gage* window enter Outlet for the "Name" and Castro Valley Outlet Gage for the "Description". Click the **Create** button to add the discharge gage to the project. In the component editor, select the Data Storage System (HEC-DSS) "Data Source" option. Navigate to the CASTRO.dss file and choose the record with the /CASTRO VALLEY/OUTLET/FLOW/16JAN1973/10MIN/OBS/ pathname using the appropriate buttons. Using the same steps as described for a precipitation gage, create a time window from 16 January, 1973 at 03:00 hours to 13:00 hours. Click the "Table" and "Graph" tabs to view the observed discharge hydrograph.

Create a paired data table for the Modified Puls routing method. Select the **Component ⇒ Paired Data Manager** menu option. Make sure the "Data Type" option is set to Storage-Outflow Functions and click the **New** button in the *Paired Data Manager* window. Leave the "Name" as Table 1 and enter a "Description" of Reach-2 in the *Create A New Storage-Discharge Function* window. Click the **Create** button to add this storage-discharge function to the project. In the component editor, set the "Data Source" to Manual Entry and the "Units" to AC-FT:CFS. Click the "Table" tab and enter the storage-discharge relationship from Table 8 (Figure 29).

Figure 29.   Storage-Discharge table for Reach-2.

Create the Basin Model

Begin creating the basin model by selecting the **Component** ⇒ **Basin Manager** menu item. Create a new basin model with a "Name" of `Castro 1` and a "Description" of `Existing Conditions`.

Create the Element Network

The Castro Valley watershed will be represented with four subbasins, two routing reaches, and three junctions. Open the new basin model map by selecting the `Castro 1` basin model in the watershed explorer. A background map can be added to the basin model by selecting the **View** ⇒ **Background Layers** menu item. Click the **Add** button in the *Map Layer Selector* screen. Navigate to the file called `castro.map` (`"hmsproj\Castro"`), which is part of the `Castro` example project. Select the file and click the **Select** button. This file is added to the "Current map layers" box in the *Map Layer Selector* window. Click **OK**.

Use the following steps and Figure 27 to create the element network:

1. Add four subbasin elements. Select the subbasin icon [icon] on the tool bar. Place the icons by clicking the left mouse button in the basin map.

2. Add two reach elements [icon].

3. Add three junction elements [icon].

4. Connect Subbasin-2 downstream to Junction-1. Place the pointer tool over the subbasin icon and click the right mouse button. Select the **Connect Downstream** menu item. Place the pointer over the junction icon and click the left mouse button. A connection link shows the elements are connected.

5. Connect the other element icons using the same procedure used to connect Subbasin-2 downstream to Junction-1. Move the hydrologic elements as necessary to complete the network shown in Figure 30. Move an element by placing the pointer over the icon in the basin model map. Press and hold the left mouse button and move the mouse. Release the left mouse button to place the icon. The upper and lower ends of a reach element icon can be moved independently.
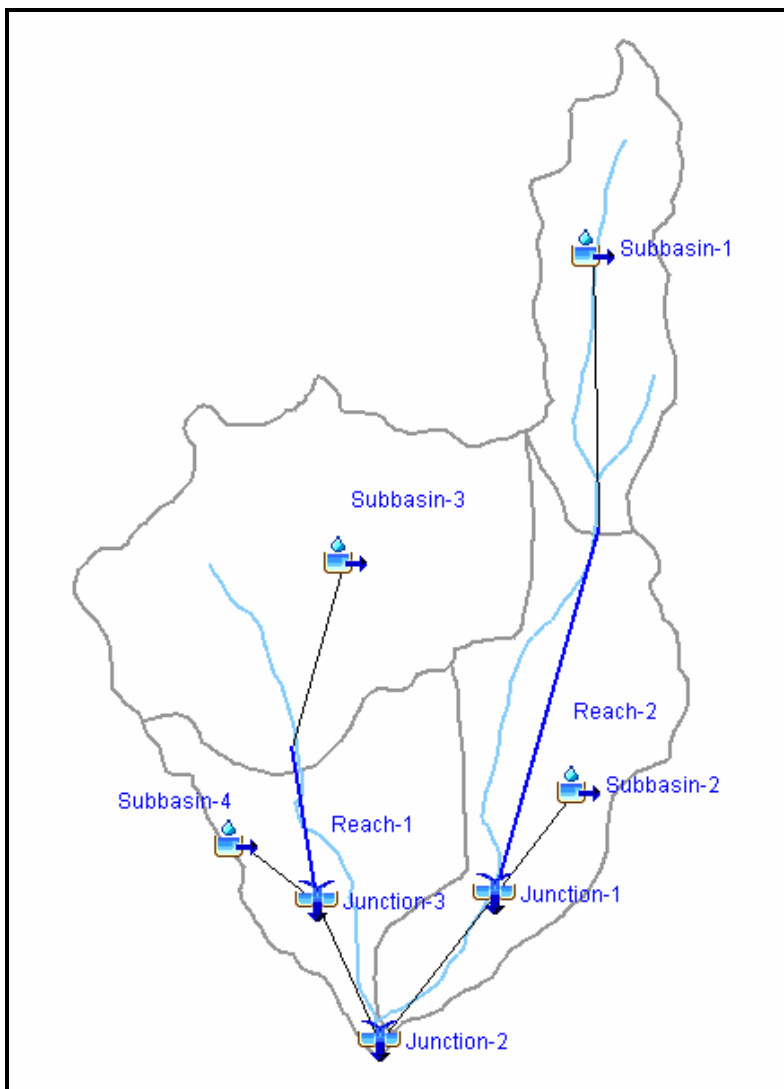
Figure 30.   Subbasin, reach, and junction elements in their correct positions.

Enter Element Data

Enter the area for each subbasin element. Select a subbasin element in the watershed explorer or in the basin map. Then, in the component editor select the "Subbasin" tab and enter the subbasin area for each subbasin element (Figure 31). Figure 31 also shows the "Loss," "Transform," and "Baseflow" tabs. One way of entering parameter data for a subbasin element is to click on each of these tabs and enter the required information. Another way to enter parameter data is to use global editors. Global editors are the most efficient way to enter data for several subbasin and reach elements that use the same methods. Select the **Parameters $\Rightarrow$ Loss $\Rightarrow$ Initial and Constant** menu item (Figure 32). Enter the loss data from Table 5 (Figure 33) and click the **Apply** button

to close the global editor. Select the **Parameters** ⇒ **Transform** ⇒ **Snyder Unit Hydrograph** menu item and enter the transform data from Table 5 (Figure 34). Select the **Parameters** ⇒ **Baseflow** ⇒ **Recession** menu item and enter baseflow data from Table 6 (Figure 35).

Figure 31.   Subbasin area.

Figure 32.   Select initial and constant global editor.

Figure 33.   Initial and constant loss global editor.

Figure 34.   Snyder transform global editor.

Figure 35.   Recession baseflow global editor.

Change the name of the three junction elements. Click the right mouse button when the
pointer is on top of the Junction-1 element name in the watershed explorer and select
Rename. Change the name to `East Branch`. Change the name of the Junction-2 and
Junction-3 elements to `Outlet` and `West Branch`, respectively.

Enter parameter data for the reach elements. Open the component editor for `Reach-1`.
Change the "Method" from `Muskingum` to `Modified Puls`. A screen appears with the
message stating that data for the old method will be lost. This message makes it more
difficult to accidentally change the method and lose parameter data. Click the "Route"
tab in the component editor and select the storage-discharge table from the drop-down
list and enter the number of subreaches from Table 7 (Figure 36). Set the "Initial"
condition to `Inflow = Outflow`. Open the component editor for `Reach-2` and enter
the data from Table 7 (Figure 37).
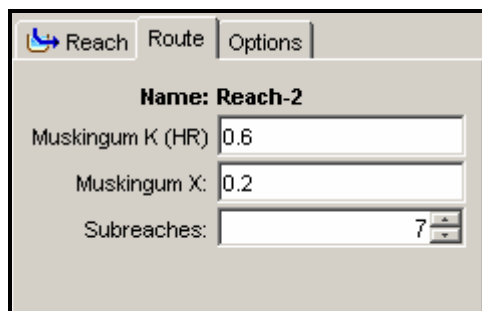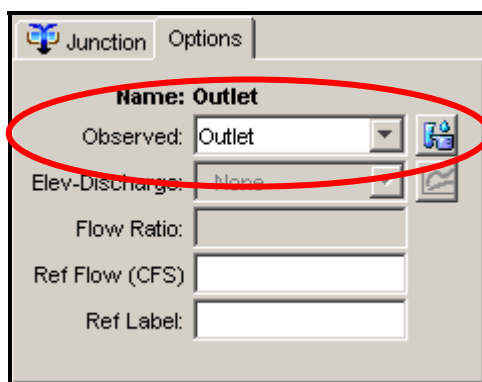
Figure 36.   Modified Puls data for Reach-1.

29

Figure 37.   Muskingum data for Reach-2.

Add an observed hydrograph to the `Outlet` element. Select this junction element in the basin map or in the watershed explorer to open the component editor. Click the "Options" tab and select the `Outlet` gage from the "Observed" drop-down list (Figure 38).

Figure 38.   Add an observed hydrograph.

The basin model is complete.

Create the Meteorologic Model

Begin creating the meteorologic model by selecting the **Component ⇒ Meteorology Manager** menu item. Click the **New** button in the *Meteorology Manager* window. In the *Create A New Meteorologic Model* window enter `GageWts` for the "Name" and `Thiessen weights, 10-min data` for the "Description". In the component editor make sure the selected "Precipitation" method is `Gage Weights` (Figure 39). At this point the watershed explorer should look similar to Figure 40.

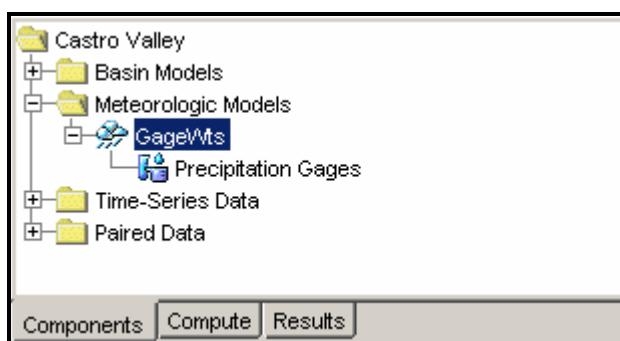_Figure 39._   _Meteorologic model component editor._

Figure 40.   _Watershed explorer showing the GageWts meteorologic model._

Subbasins need to be specified for this meteorologic model. Click the "Basins" tab in the GageWts component editor. Set the "Include Subbasins" option to "Yes" for the Castro 1 basin model (Figure 41). After this step, all subbasins in the Castro 1 basin model are added to the meteorologic model (Figure 42).

Figure 41.   _Include subbasins in meteorologic model._
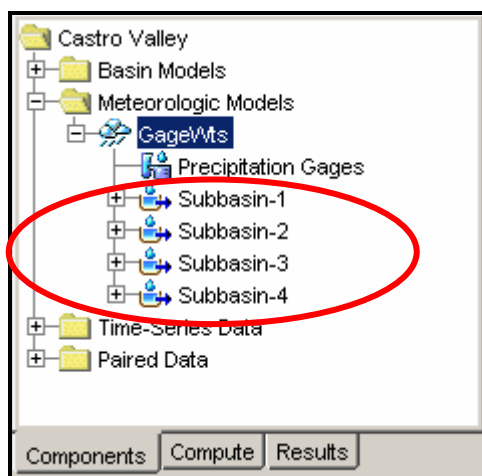
Figure 42.   Subbasins added to meteorologic model.

Use the following steps and Figure 42 to complete the GageWts meteorologic model:

1.   Add the Proctor School and Sidney School non-recording gages. Under the GageWts meteorologic model, in the watershed explorer, select the **Precipitation Gages** element to open the "Total Storm Gages" editor. Enter `Proctor` for the "Gage Name" and `1.92` for the "Total Depth". Add the `Sidney` total storm gage in the same manner (Figure 43).

Figure 43.   Proctor and Sidney total storm gages.

2.   In the watershed explorer, expand the **Subbasin-1** element and select the **Gage Weights** sub-component (Figure 44). A component editor will open with two tabs, "Gage Selections" and "Gage Weights." Depth and time weights are required for all precipitation gages with the "Use Gage" option set to "Yes". For this example, the "Fire Dept" gage will be used for all subbasin elements because it contains the storm pattern; the other gages only contain total storm depths. Once the correct precipitation gages are included for `Subbasins-1` (Figure 45), select the "Gage Weights" tab in the component editor and enter the correct "Depth Weight" from Table 9 for `Subbasin-1`. The "Time Weight" will be 1.0 for the `Fire Dept` gage in all subbasins (Figure 46). Complete this step for the remaining subbasins.
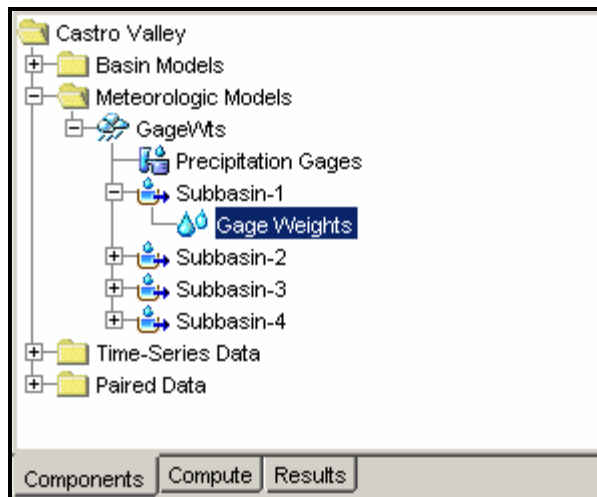
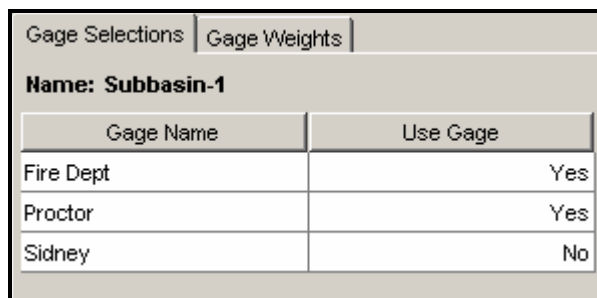Figure 44.   Selecting the Gage Weights sub-component for Subbasin-1.

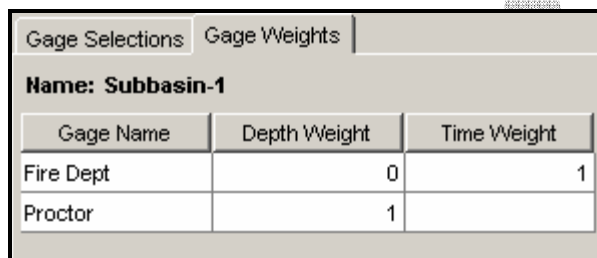Figure 45.   Selecting gages for Subbasin-1.

Figure 46.   Gage weights for Subbasin-1.

Define Control Specifications

Create the control specifications by selecting the **Components** ⇒ **Control Manager** menu item. In the *Control Specifications Manager* window, click the **New** button and enter `Jan73` for the "Name" and `16 January 1973` for the "Description." In the component editor, enter `16Jan1973` for both the "Start Date" and "End Date" (Figure 47). Enter `03:00` for the "Start Time" and `12:55` for the "End Time." Select a time interval of `5 minutes` from the "Time Step" drop-down list.

Figure 47.   Entering control specifications data.

Create and Compute a Simulation Run

Create a simulation run by selecting the **Compute ⇒ Create Simulation Run** menu item. Keep the default name `Run 1`. Select the `Castro 1` basin model, `GageWts` meteorologic model, and `Jan73` control specifications using the wizard. After the wizard closes select the "Compute" tab of the watershed explorer and click on `Run 1`. Change the description for this simulation run by entering `Existing conditions, 16 January 1973 storm` (Figure 48).

Click the right mouse button when the mouse pointer is on top of the `Run 1` name in the watershed explorer and select **Compute**. A window opens showing the progress of the compute.

Figure 48.   Simulation run component editor.

View Model Results

Begin viewing results by opening the basin model. Open the `Castro 1` basin model by clicking on its name in the watershed explorer. Select the **Global Summary Table** tool

from the tool bar to view summary results of peak flow for all elements in the basin model (Figure 49). Print the table or make a note of the computed peak discharge for Subbasin-2. View graphical and tabular results for the Subbasin-2 element. Place the pointer over the Subbasin-2 icon in the basin map and click the right mouse button.

_____

Select the **View Results** ⇒ **Graph** menu item (Figure 50). Select the **View Results** ⇒ **Summary Table** menu item to view the subbasin element summary table (Figure 51). Select the **View Results** ⇒ **Time-Series Table** menu item to view the subbasin time-series table (Figure 52).

Global Summary Results for Run "Run 1"				_ □ ×

Project: Castro Valley   Simulation Run: Run 1

Start of Run: 16 Jan 1973, 03:00   Basin Model: Castro 1
End of Run: 16 Jan 1973, 12:55   Meteorologic Model: GageWts
Execution Time: 19 May 2005, 10:14:47   Control Specifications: Jan73

Volume Units: ⊙ IN   ○ AC-FT

| Hydrologic Element	Drainage Area (MI2)	Peak Discharge (CFS)	Time of Peak	Volume (IN)
East Branch	2.38	304.28	16 Jan 1973, 07:20	0.87
Outlet	5.51	540.27	16 Jan 1973, 06:55	0.83
Reach-1	2.17	161.39	16 Jan 1973, 11:20	0.83
Reach-2	0.86	153.59	16 Jan 1973, 07:30	1.12
Subbasin-1	0.86	162.29	16 Jan 1973, 06:55	1.13
Subbasin-2	1.52	171.42	16 Jan 1973, 06:55	0.72
Subbasin-3	2.17	309.11	16 Jan 1973, 06:55	0.84
Subbasin-4	0.96	121.79	16 Jan 1973, 06:50	0.72
West Branch	3.13	243.01	16 Jan 1973, 06:50	0.80

Figure 49.   Viewing the global summary table.
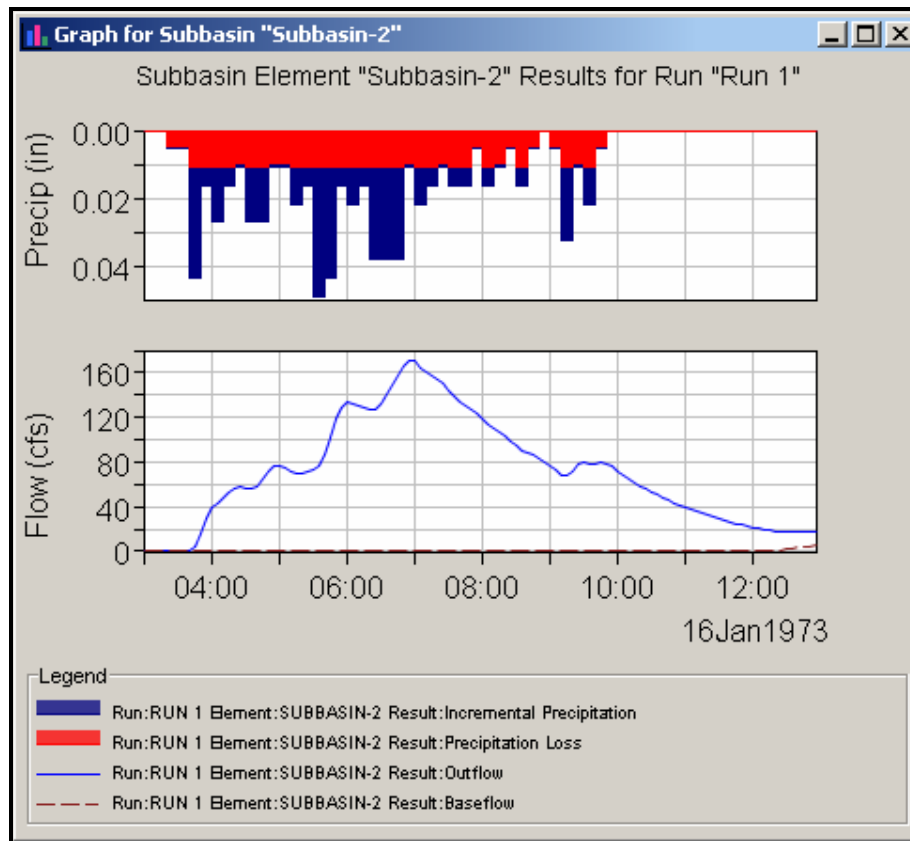
Figure 50.   Graph of Subbasin-2 results.

Figure 51.   Summary table of Subbasin-2 results.

Figure 52.   Time-series table of Subbasin-2 results.

Simulate Future Urbanization

Consider how the Castro Valley Watershed response would change given the effects of future urbanization. The meteorologic model and control specifications remain the same, but a modified basin model must be created to reflect anticipated changes to the watershed.

Create the Modified Basin Model

The urbanized basin model can be created by modifying a copy of the existing conditions basin model. Place the pointer on the Castro 1 basin model in the watershed explorer and click the right mouse button. Select the **Create Copy** option. Enter Castro 2 as the basin model "Name" and Future conditions for the "Description" in the *Copy Basin Model* window.

Modify the new basin model to reflect future urbanization. Open the component editor for Subbasin-2 (select Subbasin-2 in the watershed explorer or in the basin map). Change the percent imperviousness from 8 to 17 percent and the Snyder t_p from 0.28 to 0.19 hours.

Update the `GageWts` meteorologic model to include subbasins from the `Castro 2` basin model. Select the GageWts meteorologic model in the watershed explorer and click the "Basins" tab in the component editor. Change the "Include Subbasins" option to "Yes" for the `Castro 2` basin model.

Urbanized Simulation Run

Create a new simulation run for the future conditions basin model by selecting the **Compute ⇒ Create Simulation Run** menu item. Keep the default name of `Run 2` and select the `Castro 2` basin model, the `GageWts` meteorologic model, and the `Jan73` control specifications using the wizard. Open the `Run 2` component editor and enter `Urbanized conditions, 16 Jan 1973 storm` as the description. Compute `Run 2` and compare the peak discharges for the urbanized conditions basin model to the existing conditions basin model at `Subbasin-2, East Branch`, and `Outlet` (Table 10).

Table 10.   Peak discharges for existing and future urbanization conditions.

	Subbasin 2	East Branch	Outlet
Existing *cfs*	171	304	540
Urbanization *cfs*	211	337	580
Increase *%*	23	11	7

Results from the two simulation runs can also be compared from the "Results" tab of the watershed explorer. Results are available from the "Results" tab as long as no modifications have been made to components used by the simulation run. For example, if a constant loss rate parameter was changed in a subbasin element, then results for all simulation runs using the basin model in which the subbasin element resides will not be available. It is easy to determine if results are available. If the simulation run icon is grey, then results are not available (Figure 53) and the simulation run must be re-computed.

Figure 53. Results are not available for simulation runs.

Use the watershed explorer to compare results from `Run 1` and `Run 2`. Click the "Results" tab in the watershed explorer and select both simulation runs. The watershed explorer expands to show all hydrologic elements with results. Then, select Subbasin-2 and watch the watershed explorer expand to show all results available for this subbasin element (Figure 54). Select the Subbasin-2 "Outflow" result from the existing conditions simulation and notice the preview graph in the component editor. Hold down the Control key and select the Subbasin-2 "Outflow" result from the future conditions simulation. The hydrograph from the future condition simulation is added to the preview graph

(Figure 55). Select the **View Graph** tool bar button to open a copy of the preview graph in the desktop.
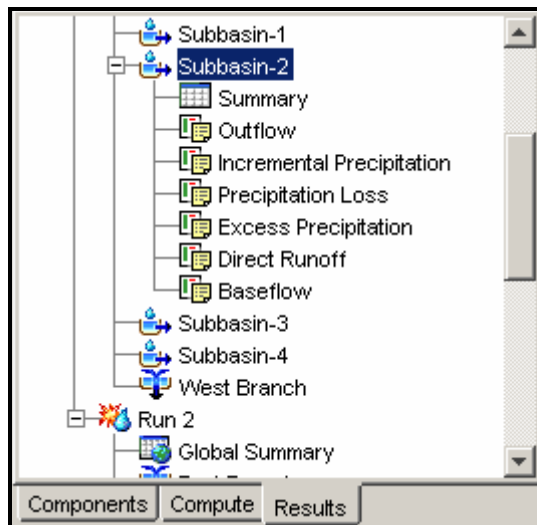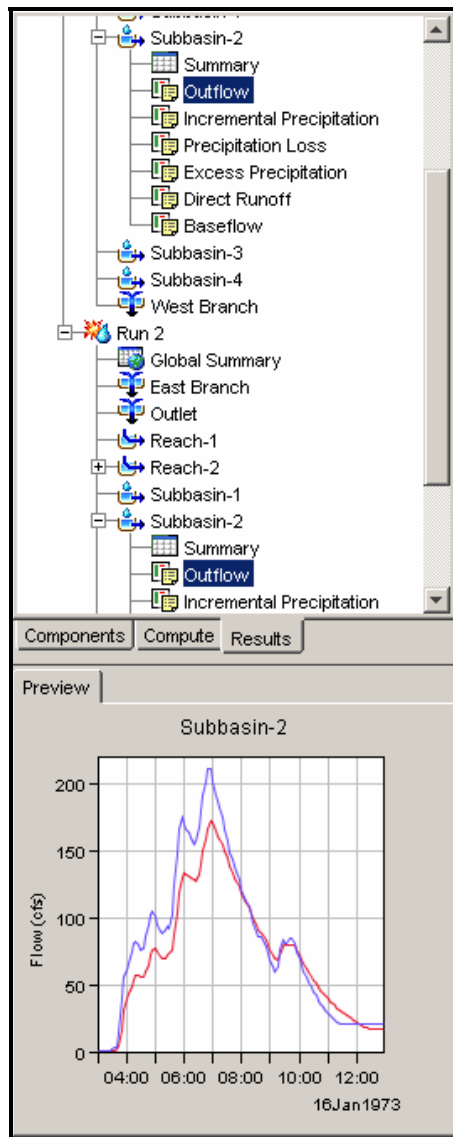
Figure 54. Available results for Subbasin-2.

Figure 55.   Comparing hydrographs from existing and future conditions simulations.

Save the project by selecting the **File ⇒ Save** menu item. The example application is now complete.

Appendix

Create and Compute an Optimization Trial

Before an optimization trial can be created, a simulation run using a basin model with observed flow must exist. An optimization trial is created by selecting the **Compute ⇒ Create Optimization Trial** menu option. A wizard steps the user through the process of creating an optimization trial. First, a name must be entered, then an existing simulation run must be selected, and finally a hydrologic element containing observed flow must be selected. The new optimization trial is added to the "Compute" tab of the watershed explorer (Figure A1). In the component editor, the user can enter a "Description", change the simulation run used by the optimization trial, and select the search method used to find optimal parameter values. Also, the user has the option of changing the tolerance and the number of iterations to control when the search for optimal parameter values ends.

Click the plus/minus box next to the optimization trial name to expand or collapse the watershed explorer. Select the **Objective Function** element in the watershed explorer to add a new editor to the optimization trial component editor (Figure A2). On this editor the user can select the objective function from the "Method" drop-down list and change the location used for comparing observed and simulated hydrographs. In addition, start and end dates and times can be edited.

An optimization trial requires hydrologic element parameters. To add a parameter, click the right mouse button when the pointer is on top of the optimization trial's name in the watershed explorer and select **Add Parameter** (Figure A3). A new element is added to the watershed explorer with the name "Parameter 1." Figure A4 shows the editor for this new element. From the parameter tab of the component editor, the user selects the hydrologic element and the parameter for that element. This parameter is adjusted during the optimization trial in an attempt to find a value which minimizes the difference between simulated and observed hydrographs. The user has the option to select a different initial value for the parameter, enter minimum and maximum value constraints, and select whether the parameter is locked during the optimization trial. More than one parameter can be added to an optimization trial.

An optimization trial can be computed from the **Compute** menu or from the watershed explorer. Results for an optimization trial are only available from the "Results" tab of the watershed explorer. Additionally, results only exist for hydrologic elements located upstream of the element containing the observed hydrograph used in the optimization trial.
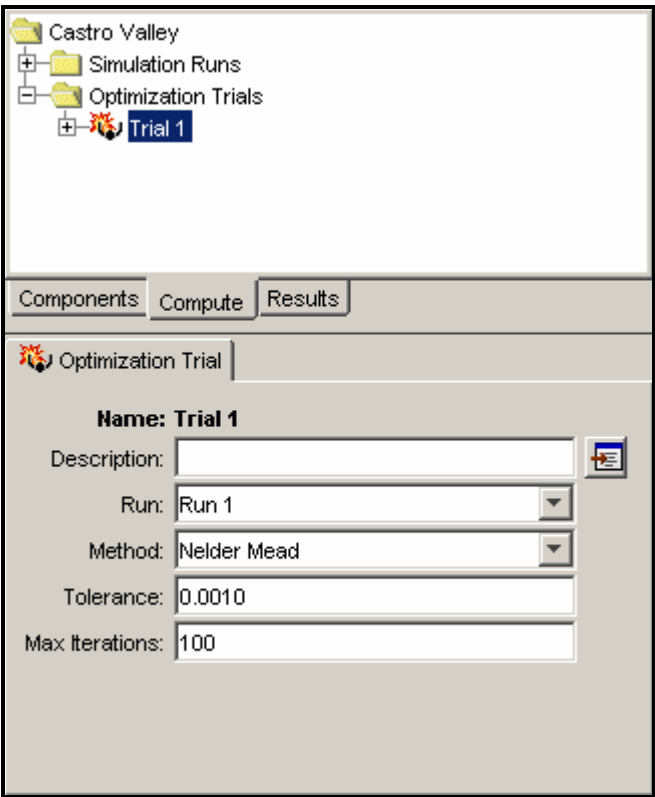
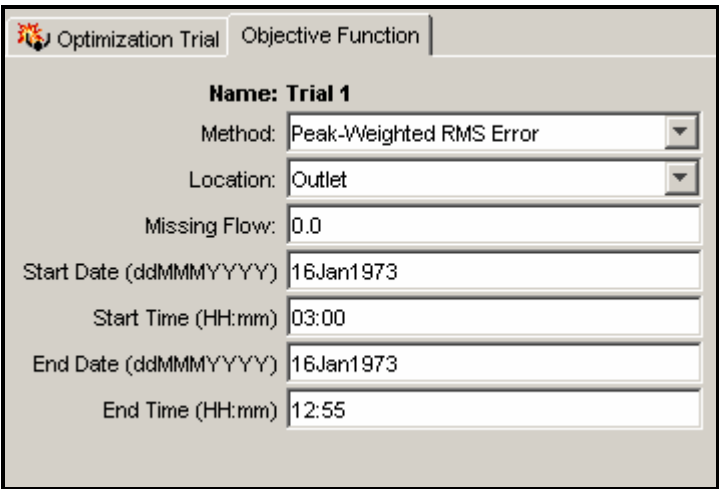*Figure A1.   Optimization trial.*

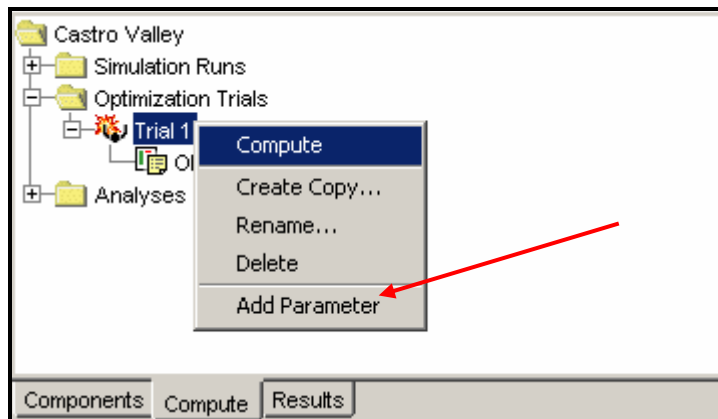*Figure A2.   Objective function editor.*

_____

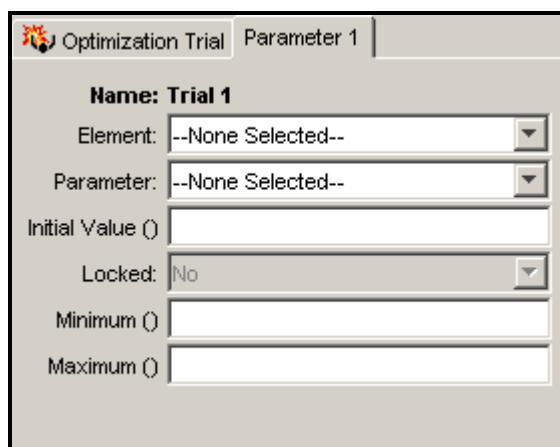*Figure A3.   Add a parameter to an optimization trial.*

Figure A4.   Parameter tab (The editor updates when changing the "Element" and "Parameter" options).

Create and Compute a Depth-Area Analysis

Before a depth-area analysis can be created, a simulation run using a frequency storm meteorologic model must exist. A depth-area analysis is created by selecting the **Compute** ⇒ **Create Analysis** menu option. A wizard steps the user through the process of creating a depth-area analysis. The user must choose the analysis type in the first step of the wizard. Currently, the only option is **Depth-Area**. In the next step, the user must enter a name for the depth-area analysis. In the last step, the user must select an existing simulation run. The analysis will use the basin model, meteorologic model, and control specifications from the chosen simulation run. The new depth-area analysis is added to the "Compute" tab of the watershed explorer (Figure A5). A new analysis can also be added to a project using the analysis manager. In the component editor, the user can enter a "Description" and change the simulation run used by the depth-area analysis.

A depth-area analysis requires the user to select points, hydrologic elements, in the basin model where outflow from a frequency event is needed. The depth-area analysis automatically adjusts precipitation depths in a frequency storm meteorologic model to reflect the upstream drainage area for each analysis point. The user selects points of interest (analysis points) on the "Analysis Points" tab of the component editor (Figure A6). Click the mouse in the first row of the "Element" column and a drop-down list will appear containing hydrologic elements in the basin model.

A depth-area analysis can either be computed from the **Compute** menu or from the watershed explorer. Results for a depth-area analysis are only available from the "Results" tab of the watershed explorer. Additionally, results are only available for hydrologic elements selected on the "Analysis Points" tab of the depth-area analysis component editor.
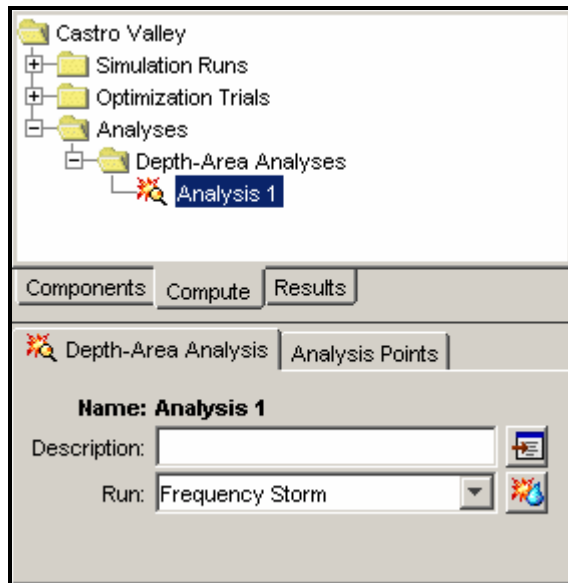

Figure A5.   Depth-area analysis component editor.

Figure A6.   Selecting analysis points for a depth-area analysis.